MICHAEL D. GRIFFIN

COMPUTATION FLUID DYNAMICS – A TOOL FOR MISSILE DESIGN

The development of high-speed computers and algorithms for the solution of systems of partial differential equations has made practical the analysis of many complex fluid flows. This capability allows more efficient design and analysis of aerodynamic vehicles, including tactical missiles. Computational analysis vastly extends the useful range of theoretical studies and allows unpromising initial designs to be rejected without the need for more expensive wind tunnel and flight experiments.

INTRODUCTION

The design of a tactical rocket or ramjet missile and the determination of its performance are dependent on an understanding of fluid dynamic phenomena. Propulsive forces are derived from the manipulation of fluid streams with highly complex flows in the combustion chamber and exhaust nozzle, while flight performance and guidance and control capability are governed by the external aerodynamic configuration. An airbreathing missile such as a ramjet will have an air inlet that must operate at high efficiency and in which very slight changes in configuration can completely alter the overall performance. The missile structure must be designed to withstand predicted aerodynamic loads. Aerodynamic heating determines the design of effective thermal protection systems for the missile structure.

Analysis of these aspects of its performance, therefore, requires the ability to analyze the motion of air and other gases in and around the vehicle. Within the last two decades, the capabilities of computers and the sophistication of numerical methods have grown to the point where many classically important and previously unsolved problems can be treated routinely. The term "computational fluid dynamics" is used to set this field apart from others.

Computational fluid dynamics as a specialty area within aerospace engineering is a new and dynamic field. Consequently, most of the effort in that area has been to develop new algorithms and new codes, and to try them on new and different problems. This approach is natural and appropriate for research. With few exceptions, however, there has been little effort to develop codes that are working tools capable of supporting more general design and analysis efforts. Far more computer runs are made to demonstrate an algorithm or code than are made to obtain needed data. Furthermore, most available codes cannot be used successfully without the supervision of a specialist (often the investigator who designed the code). Computational fluid dynamics has not reached the point where library routines are available for general use, e.g., in a discipline such as control theory.

This situation is changing as groups at various laboratories are developed to provide support to other activities to the extent allowed by the state of the art. Such is our approach to the problems of tactical missile design at APL. There is a substantial body of established techniques that could be useful in this area. Analytical methods can be used by a skilled engineer to weed out unpromising designs, or to optimize promising ones, at considerably less expense than relying on experiments for all results.

In this article, we will discuss the difficulties involved in analyzing fluid dynamic systems and will show how numerical methods can be employed in such tasks. We will consider some examples of the application of computational techniques in supersonic missile inlet design and will examine the direction of possible future efforts in this field.

EQUATIONS OF GAS DYNAMICS

The basic equations of fluid mechanics have been known for well over a century. They consist of a set of partial differential equations expressing conservation of mass, momentum, and energy. In vector form, these are, respectively,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \mathbf{V} = 0 , \quad (1)$$

$$\frac{\partial \mathbf{V}}{\partial t} + \mathbf{V} \cdot \nabla \mathbf{V} = - \frac{1}{\rho} \nabla : \tau , \quad (2)$$

and

$$\frac{\partial e}{\partial t} + \mathbf{V} \cdot \nabla e = \frac{1}{\rho} \nabla \cdot k \nabla T + \tau : \nabla \mathbf{V} , \quad (3)$$

where ρ is the fluid density, V is the vector velocity,.e is the internal energy, and T is the absolute temperature. The stress tensor τ includes the effects of viscosity.

A variety of auxiliary equations are necessary to provide equations of state and constitutive relationships appropriate to particular classes of fluids. For example, an ideal gas has the state equation

$$p = \rho RT, \tag{4}$$

where p is the pressure and R is the specific gas constant. A calorically perfect gas relates internal energy, e, to temperature as

$$e = c_V T, \tag{5}$$

where c_V is the constant volume heat capacity. A linear or Newtonian fluid is one in which the internal viscous stress forces are linearly proportional to the rate of strain in the fluid. This constitutive relationship is directly analogous to that for a linear or Hookean solid, where stress is proportional to strain itself.

These equations, together with particular sets of initial and boundary conditions, are sufficient to describe all flows in which the fluid may be regarded as a continuous medium. However, they are so complicated that no general solutions are known, and only about 80 particular solutions are known that satisfy the full set of equations for some special geometry. While some of these are quite instructive, it cannot be said that they apply directly to many systems of engineering interest.

Flowfield analysis for missile systems is complicated further by the presence of complex chemical reactions of liquid droplets or solid particles in the combustion chamber and by high-temperature effects on the external flow. Under these conditions, no rigorous analytic solutions are possible. Finally, under most typical flight conditions one can expect turbulent rather than laminar flow. Exact analysis of turbulent flows is impossible with present mathematical techniques.

APPROXIMATIONS IN FLUID MECHANICS

The great complexity of the full set of governing fluid dynamics equations has meant that, in practice, nearly all useful results are obtained with approximate analyses, where the physics of the situation appear to justify the neglect of certain terms. Thus we may note that the flow of air at speeds of less than 100 meters per second at room temperature will result in density changes of less than 5%. Such "low speed" flows are then approximated as incompressible, or constant density, an assumption that greatly simplifies the equations of motion. In such cases our mathematics would make no distinction between the flow of air and that of water, a seemingly absurd procedure. Yet incompressible flow theory can be used to provide excellent predictions of airfoil lift at low speeds, as an example.

One can encounter some surprising results with this approach, however. For example, air has a vanishingly small viscosity compared to fluids such as oil or even water. It thus seems an excellent approximation in aerodynamics to neglect the viscous or friction terms in the equations of motion. When this is done, a variety of interesting and important solutions become available, allowing predictions of such diverse phenomena as shock wave propagation, rocket engine flowfields, airfoil lift at small angles of attack, etc. However, inviscid theory predicts zero aerodynamic drag for a body moving at subsonic speed through the air. This is clearly not the case and, in fact, many bodies such as bullets or golf balls encounter far greater resistance than simple air friction would produce. The resolution of this paradox (credited to the French mathematician d'Alembert) lies in the development of boundary layer theory by the German scientist Prandtl. This theory shows that the correcting limiting case for infinitesimal viscosity is not the vanishing of frictional effects but rather the concentration of these effects in a very small layer near the surface of a body. Under some circumstances this layer can alter completely the structure of the flowfield at large as compared to that predicted by inviscid theory.

Thus theoretical results derived on the basis of "obviously" reasonable physical approximations may be very wrong, and there is a justifiably strong dependence on experiment and prior art by engineers engaged in the design and development of aerodynamic vehicles. However, with wind-tunnel cost in the range of $1000 per hour (not including model building and set-up costs) test programs can be quite costly. Moreover, many situations of interest cannot be duplicated in an experiment. A recent example is found in the space shuttle, where testing to determine peak heating rates during reentry cannot be done in ground-based facilities at full-scale flight conditions.

This is not to say that computational methods can solve all problems. Just as there are physical situations not easily modeled by experiment, so there are those where the appropriate mathematical model is not known. Combustion phenomena in particular may elude analysis if the reaction mechanism is unclear. Turbulent flows, particularly at high speed, have so far resisted attempts to develop general methods for their simulation. An appropriate combination of experiment and analysis is needed.

ROLE OF THE COMPUTER

As in so many areas of science and engineering, the rapid progress in digital computer development over the last two decades has revolutionized the thinking of practitioners in fluid dynamics.[1] The computer has allowed numerical analysis to augment the traditional areas of theory and experiment. The basic idea in computational fluid dynamics is to approximate the continuous domain and boundaries of interest by a network or grid of points spaced closely enough to represent the flow adequately (not necessarily an easy task!). The partial derivatives that appear in the governing equations are then replaced by algebraic expressions or procedures that are intended to approximate, on the finite set of grid points, the true derivatives. This then yields a set of algebraic equations that are solved for the flowfield variables.

There are many methods for approximating the derivatives involved and for solving the resulting system of algebraic equations. However, some seemingly obvious approaches can exhibit substantial inaccuracy or unstable behavior under certain conditions. It is a goal of numerical analysis to develop stable, accurate methods that can be programmed easily and that will run efficiently. This is a topic of continuing research interest for systems of partial differential equations in general and the governing equations of fluid dynamics in particular. However, for "routine" engineering use, the field is dominated by explicit finite difference methods.

A simple example of an explicit finite-difference method is furnished by considering the one-dimensional heat equation

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2} ,$$

where

$$\begin{aligned} T(0,t) &= T_0 \\ T(L,t) &= T_L \\ T(x,0) &= \text{known} \\ \rho &= \text{density} \\ c &= \text{specific heat capacity} \\ k &= \text{thermal conductivity} \\ \alpha &= k/\rho c , \end{aligned} \tag{6}$$

which could be used to describe the flow of heat in a bar (Fig. 1) as a function of length, L, and time, t, given appropriate initial and boundary conditions. Let

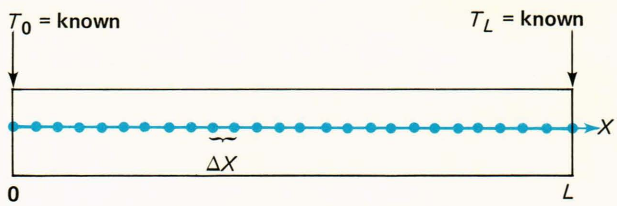


Figure 1 – One-dimensional heat conduction.

$$\begin{aligned} t_n &= n\Delta t \\ x_i &= i\Delta x,\ i = 0, 1, 2, \ldots , L/\Delta x \\ T_i^n &= T(x_i, t_n) \end{aligned} \tag{7}$$

for notational convenience. A possible finite-difference representation for this equation is

$$\frac{T_i^{n+1} - T_i^n}{\Delta t} = \alpha \frac{T_{i+1}{}^n - 2T_i^n + T_{i-1}{}^n}{\Delta x^2} . \tag{8}$$

Since it is desired to find temperatures in the bar as time increases, we solve to yield

$$T_i^{n+1} = T_i^n + \frac{\alpha \Delta t}{\Delta x^2} \left(T_{i+1}{}^n - 2T_i^n + T_{i-1}{}^n \right) , \tag{9}$$

which is valid for all values of the index i except the first and last points, where the differencing scheme clearly breaks down. However, the end points are specified by boundary conditions (not a trivial problem), so there is no ambiguity in the scheme.

Notice that the new temperatures at all points are explicitly available in terms of old temperatures, hence the term "explicit scheme." But this is only one possible finite-difference representation of the original equation; we could as well do the spatial differencing at the new time level to obtain

$$\frac{T_i^{n+1} - T_i^n}{\Delta t} = \alpha \frac{T_{i+1}{}^{n+1} - 2T_i^{n+1} + T_{i-1}{}^{n+1}}{\Delta x^2} . \tag{10}$$

We rearrange to yield

$$\begin{aligned} &\left[1 + \frac{2\alpha\Delta t}{\Delta x^2}\right] T_i^{n+1} = \\ &T_i^n + \frac{\alpha\Delta t}{\Delta x^2} \left(T_{i+1}{}^{n+1} + T_{i-1}{}^{n+1} \right) , \end{aligned} \tag{11}$$

which shows that T_i^{n+1} is given in terms of T_i^n as well as the unknowns $T_{i+1}{}^{n+1}$ and $T_{i-1}{}^{n+1}$. Since the above equation can be written at each grid point in the bar (excepting again the boundaries, which are known), we obtain a system of simultaneous linear algebraic equations wherein each T_i^{n+1} is implicitly known in terms of others. Efficient means are available to solve such systems, yielding the entire set of T_i^{n+1} at once.

Explicit schemes are more easily programmed and generally quicker to run for each step than implicit schemes. However, explicit schemes will always have a limiting step size that may be used, and this step size may be smaller than that actually required to maintain good accuracy. Implicit schemes will not have a limiting step size, or will have a larger limit, and thus may require fewer steps. One may, however, not know what constitutes an appropriate step size for an implicit method. Explicit schemes tend either to run successfully, producing good results, or to fail completely, whereas implicit schemes will often continue to run even when they produce erroneous results. It is for these reasons that explicit schemes have been most favored in engineering applications, although considerable recent progress has been made in the development of implicit methods.[2,3]

For the more complicated equations of fluid dynamics, the situation is similar; it is merely required to perform, in parallel, identical operations on each equation so as to advance the whole set together. One may consider that the dependent variable is a vector in this case.

In the example above, we considered a transient problem, the time evolution of temperature distribution in a bar with fixed end conditions. Carried out in time, the solution should, and will, render the correct steady-state linear distribution. This is unimportant for the conduction equation; if a steady-state problem is to be considered (e.g., temperature distribution in a flat plate with fixed edge temperatures), there are efficient methods to find the solution without iteration to a steady state. But with the more complicated coupled systems of equations in fluid mechanics, this turns out not to be the case. Some steady flows can be solved as such, but many others can be obtained only by iterating the full time-dependent equations to find the limiting behavior at large times. The full Navier-Stokes equations are in this class, but the inviscid equations may or may not be. Supersonic flows can be solved using less costly steady-state methods, whereas subsonic flows cannot. The solution of high-speed, steady inviscid flows is therefore relatively efficient and is also the appropriate starting point for many problems relevant to tactical missiles.

The use of finite-difference methods to integrate systems of partial differential equations numerically is not new. Prior to the development of high-speed computers, few projects could justify the enormous effort involved in obtaining a solution by hand calculation. One obvious exception was the development of the atomic bomb during World War II. Work by von Neumann and others during this period was crucial in establishing the basics of stability theory and in initiating the development of digital computers. Ulam[4] has some interesting comments on this extremely dynamic period, while Roache[5] provides an overall survey of the field up to about 1970.

APPLICATIONS TO MISSILE DESIGN

The state of the art in computers and algorithms does not yet allow the comprehensive calculation from first principles of external or internal flowfields pertinent to tactical missile performance. With the largest computers available, this goal may be approached for cases with fairly simple geometry or combustion models.[6,7] However, in no sense are the codes suitable for cost-effective use as a design tool. Fortunately, much useful engineering analysis can be accomplished without resorting to the most faithful and expensive computations possible.

From an engineering viewpoint, the most important external flowfield information is the pressure distribution. This allows calculation of the supersonic wave drag on the missile and the lift and moment forces that must be manipulated to guide the vehicle. As long as flow separation does not occur, the pressure field is extremely well predicted using inviscid calculations. Efficient and cost-effective three-dimensional Euler equation codes are now fairly common and are capable of handling realistic wing-body geometries with control surfaces.[8,9] Aerodynamic heating and skin friction effects cannot be computed because viscosity is neglected. However, boundary layer codes exist that can yield such information, using as input data the missile surface conditions as computed by the inviscid codes.

At high angles of attack, separated flow is to be expected on the leeward side of the missile body. In that case, the approach outlined above becomes invalid. The only way to simulate such flows adequately is to use the full Navier-Stokes equations in a time-iterative manner to allow relaxation to the steady-state flow. This is expensive and, as indicated above, not in the class of practical design tools for multi-dimensional problems.

Supersonic inlet design is another application area for computation fluid dynamics techniques in tactical missile design and is the main area of concentration in our work. The purpose of the inlet is to capture and compress a stream of high-speed air and direct it to a combustion chamber with as high an efficiency as possible. This translates into minimizing losses caused by shock waves and viscous effects. There are nontrivial problems involved in starting the flow through such inlets, and in operating them so as to prevent "unstarts." Additionally, conditions in the combustion chamber can react back on the inlet, usually unfavorably, by means of feedback through the subsonic boundary layer. For these and other reasons, inlet design has been traditionally an area where experience and intuition have been of paramount importance. It is a ripe field for the introduction of analytical support.

We have for several years been examining various aspects of a new propulsion concept, the dual-combustion ramjet,[10,11] as shown in Fig. 2. Externally compressed air is subdivided such that a small fraction is ducted to a subsonic combustor. All the fuel is added in the subsonic combustor, which acts as a fuel-rich, hot-gas generator for the supersonic combustor ($M > 1$). The major portion of the air bypasses the gas generator and is ducted through an air conduit to the supersonic combustor (at $M = 1$), where it mixes and burns with the exhaust of the gas generator. The final shocks shown in the supersonic air duct result from the combustion-induced pressure disturbances generated by the mixing and heat release processes.

The approach to the preliminary design of the supersonic inlet for this missile represents our first extensive use of computational fluid dynamics techniques as a tool in the design process. Because (as noted) we cannot achieve a brute-force solution to

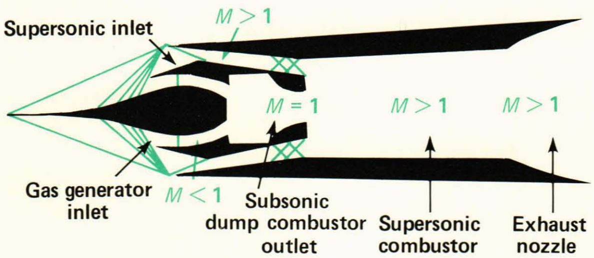


Figure 2 – Dual-combustion ramjet configuration.

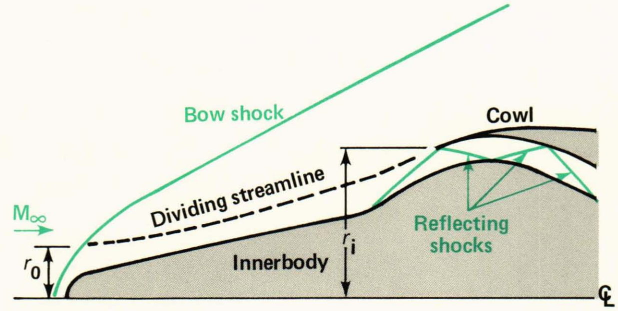


Figure 3 – Hypersonic inlet and shock system at a flight Mach number M below the inlet design Mach number M_d. When $M = M_d$, the bow shock will interesect the cowl lip, and $r_0 = r_i$.

the full problem from first principles, our goal has been to apply more limited techniques within their range of validity. The analysis is used with the goal of eliminating the poorer designs as candidates for wind-tunnel tests and optimizing certain aspects of promising designs. We do not expect with currently available methods to produce fully accurate performance assessments in all areas. This remains the province of experiments and flight tests.

As an example, consider an annular inlet as shown in one-quarter section in Fig. 3. The innerbody compresses the flow entering the cowl, which then turns the flow back toward the combustion chamber. A system of reflecting shocks will form in the annular air passage, as shown. In general, the stronger the shocks the less efficient the inlet. Figure 4 is an inviscid calculation of the duct flow, showing the shock system, the total pressure, and the local Mach number along the center streamline, $\psi = 100$. The first few shocks, which must turn the flow, are strongest. The strength of the entrance shock is controlled by the initial upper cowl angle seen by the entrance flow. Higher cowl angles, up to a point, produce lower overall shock-train losses. This effect is shown in Fig. 5, where the kinetic energy efficiency, η_{KE}, at the combustion chamber entrance is plotted against Mach number for three cowl angles. Shock waves produce losses in total pressure, decreasing the momentum of the air and lessening the ability to produce thrust. The internal kinetic energy efficiency is a measure of these losses. The relationship between η_{KE} and the mass-averaged total pressure ratio $p_{t_\infty}/p_{t_{exit}}$ in terms of M_∞ and the ratio of specific heats, γ, is shown in the figure. The analysis enables us to locate the shocks, isolates those that produce the greatest losses, and provides a quantitative measure of the effect of altering the inlet design to adjust these shock waves.

However, the internal flow loss is not the whole story, because higher cowl angles tend to produce the more external wave drag, as shown in Fig. 6. Here, the cowl wave drag coefficient $C_{D_{wave}}$ is defined as the integral of the pressure acting on the projected area of the cowl in the axial direction $A_\perp$ normalized by the product of the free-stream dynamic pressure q_∞, and the projected frontal area of the inlet, A_{max}. Curves are shown for two inlet design Mach numbers, $M_d = 6$ and 7. Thus the designer must choose an appropriate compromise between internal efficiency and external drag. Modern computational

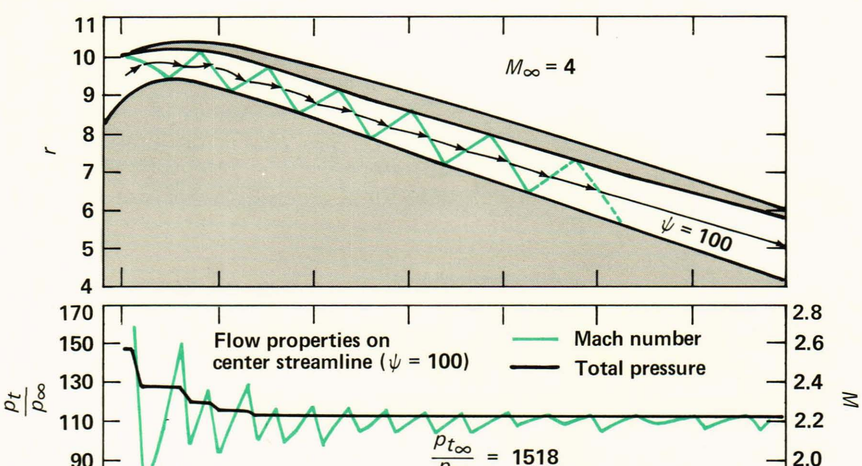
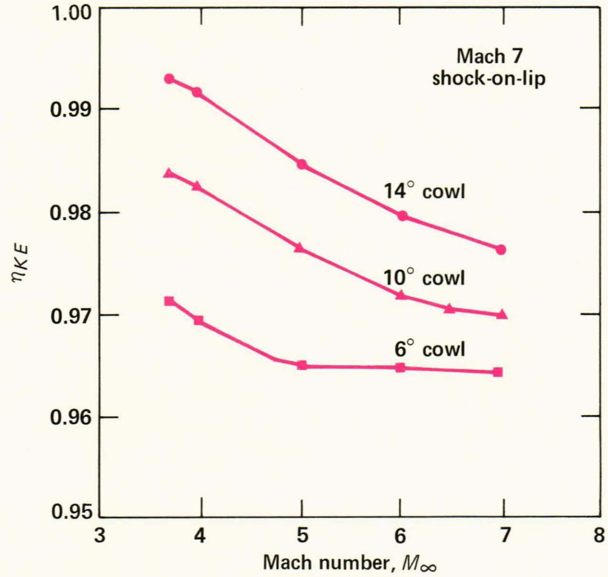


Figure 4 – Shock structure for internal flow.



$$\eta_{KE_{\text{mass avg}}} = 1 - \frac{2}{(\gamma-1)\,M_\infty^2}\left[\left(\frac{p_{t_\infty}}{p_{t_{\text{exit}}}}\right)^{\frac{\gamma-1}{\gamma}} - 1\right]$$

Figure 5 – Kinetic energy efficiency versus Mach number.

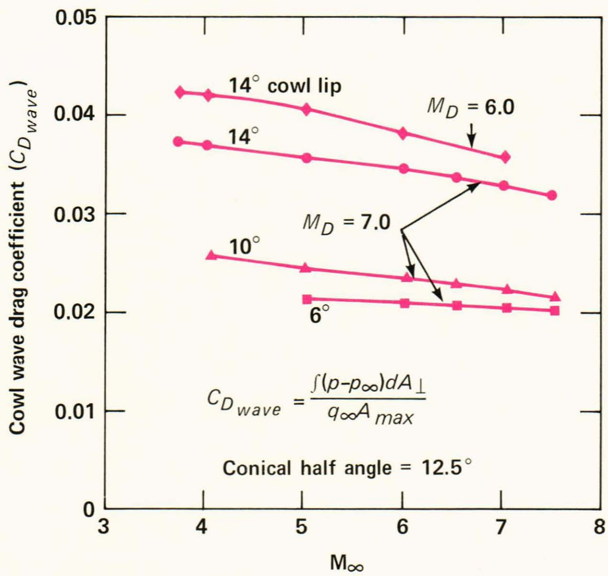


Figure 6 – Cowl wave drag versus Mach number.

fluid dynamics techniques make such trade-off analyses possible, indeed routine, at very reasonable cost.

Once a family of inlet designs has been examined and those with higher efficiency have been identified, it becomes necessary to examine the effects of viscosity. The numerical techniques required to obtain solutions to flows with viscosity are formidable and require about an order of magnitude larger computational effort. A discussion of the procedures developed at APL to solve this more difficult problem will not be given. Suffice it to say, though, that the flow in the internal passages shown in Fig. 4 becomes a fully developed viscous flow at typical inlet operating conditions. This does not diminish the value of the inviscid analyses in eliminating relatively low efficiency designs from consideration, but it shows that a more sophisticated analysis and wind-tunnel testing are required to select a final design and determine its true performance.

The above example illustrates where, once the fluid dynamic analysis is available, an assessment of the design in terms of internal efficiency can be easily made. In other cases, sophisticated post-processing of the computed flowfield may be required in order to obtain values for parameters of design interest.

Consider Fig. 3 again. The indicated dividing streamline separates the ingested airflow from that which goes around the cowl. For an axisymmetric body, this streamline is actually a portion of a streamtube called the capture streamtube, which retains its identity (though not its rotational symmetry) for angle-of-attack flight when the flow becomes fully three-dimensional. The capture streamtube intersects the bow shock generated by the missile along a locus of points called the capture envelope. The projected area of this envelope in a plane normal to the wind axis defines the air capture area A_0. Knowledge of the air capture of the inlet is essential in determining overall missile thrust. These data can be obtained analytically for conical inlets at zero angle of attack, but for more complex shapes or flight at angle of attack the problem is intractable.

Figure 7 shows the result for a pair of conical inlet designs at various Mach numbers and angles of attack, α. To get this information, the flowfield data are saved as the calculation progresses from the missile nose back to the inlet cowl. Streamlines are traced backward from the cowl to their intersections with the shock, then projected along the wind axis to a common reference plane. The capture envelope in this plane is then integrated to yield A_0.[12]

An interesting point is that to a researcher in computational methods, a problem is solved when the flowfield is obtained, but to an engineer using computational fluid dynamics as a design tool, much work may be required in processing the flowfield to obtain various parameters of interest, such as η_{KE}. For the example above, the post-processing was as expensive as the original computation, and much more difficult to develop. This is a theme that runs through much of our work in attempting to adopt modern computational techniques for use as design aids.

PARABOLIZED NAVIER-STOKES SOLUTIONS FOR VISCOUS INLET FLOWS

Purely inviscid analyses have great utility, but also possess a number of limitations. To illustrate, consider again the internal duct flow of Fig. 4. Viscous effects are important in a configuration such as this, especially toward the end of the duct where a fully developed viscous flow could be expected. Since the internal efficiency will always decrease in the presence of viscosity, it is of interest to know where such effects dominate. The invisicid analysis offers no clue. It is, of course, possible to use the inviscid flow in the classical manner to provide edge conditions for a boundary layer code, and when this is done we find that the internal flow becomes fully developed about three-fourths of the way through the duct. At this point, the concept of a boundary layer becomes ill-defined, as there is no inviscid core flow to provide edge conditions. This type of analysis cannot then be used to obtain quantitative efficiency results. A fully viscous code is required.

Another area of concern is external flow separation at angle of attack. At high speeds and high altitudes, leeside flow separation can occur at angles of attack possibly as low as 5° to 10°. This separation can strongly affect air capture, additive drag, and η_{KE}. The flowfield in such a case can be extremely difficult to obtain; however, in our work it is often not necessary. What is needed is knowledge of the conditions under which separation will occur. This defines the angle of attack boundary that is operationally useful. Again, a viscous analysis is required to obtain an indication of flow separation.

One technique for obtaining solutions for fully viscous flows is to solve the equations for unsteady flow, beginning with a set of specific initial conditions and seeking convergence to a steady-state result. These so-called "time dependent" techniques (see for example Refs. 6 and 7) require run times on very large computers measured in hours. From the point of view of cost considerations these methods are impractical as design tools. Consequently, a somewhat approximate but considerably less expensive method has been adopted. It is based on a parabolization of the Navier-Stokes equations developed by Schiff and Steger.[13-15] The PNS code was designed originally for external flow applications, whereas our primary area of concern was to obtain estimates of losses due to internal viscous effects. Accordingly, the code was adapted and extended to handle internal flow cases, and the results were compared to those obtained inviscidly.[16]

Solutions have been generated for two-dimensional and axisymmetric inlet configurations at a variety of in-flight free-stream conditions and wind-tunnel test conditions. Mach numbers and pressure profiles have been compared with inviscid results. Boundary layer displacement thickness calculated using the PNS solutions were used to adjust the compression surfaces of a wind-tunnel model to account for the mass flow defect caused by the viscous effects near the walls as compared to the duct geometry designed on the basis of inviscid flow calculations. Subsequent test data show very good agreement between the PNS code predictions and experiment.

Figures 8 and 9 compare Mach number and pressure profiles for the PNS and inviscid codes at Mach 4 for a particular inlet configuration of interest, similar in general form to that in Fig. 3. The Mach number profiles are taken at the cowl lip, the diffuser throat, mid-station, and exit. Very good agreement with the inviscid results is observed in the core flow,

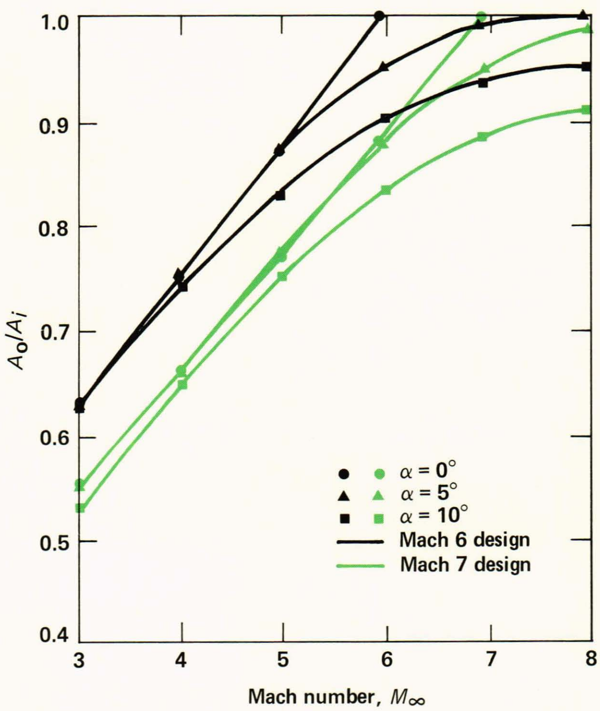


Figure 7 – 12.5° cone air capture versus Mach number.

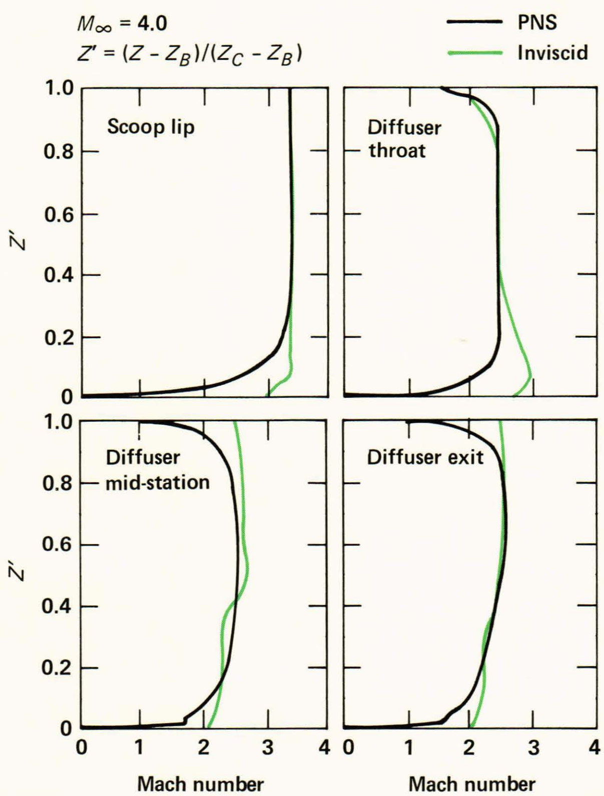


Figure 8 – Mach profiles for an axisymmetric supersonic inlet internal flowfield.

and the boundary layer is clearly evident near the walls in the viscous analysis. In Fig. 9, the pressure profiles are taken at the cowl lip, slightly aft of the diffuser throat, and at the exit. Agreement again is good except at the exit station, where the PNS results do not predict the large midstream pressure peak that is present in inviscid core solutions. As indicated previously, there is no longer an inviscid core at this point, and thus little reason to expect agreement between the two analyses. Note also that viscous effects serve to damp the pressure oscillations seen in the inviscid results.

Boundary layer displacement thickness, δ^*, is plotted in Fig. 10 as a function of axial station for both the inner and outer walls. The fluctuations in δ^* closely correlate with fluctuations in pressure caused by the reflecting internal shocks. Similar fluctuations are observed in calculations made with the more approximate superposition technique where we employed a simplified boundary layer calculation using conditions at the boundary layer edge provided from the inviscid analysis. For constructing wind-tunnel models or ultimately in the design of engines for flight, a smooth curve is drawn through the δ^* data and the surfaces are adjusted correspondingly to

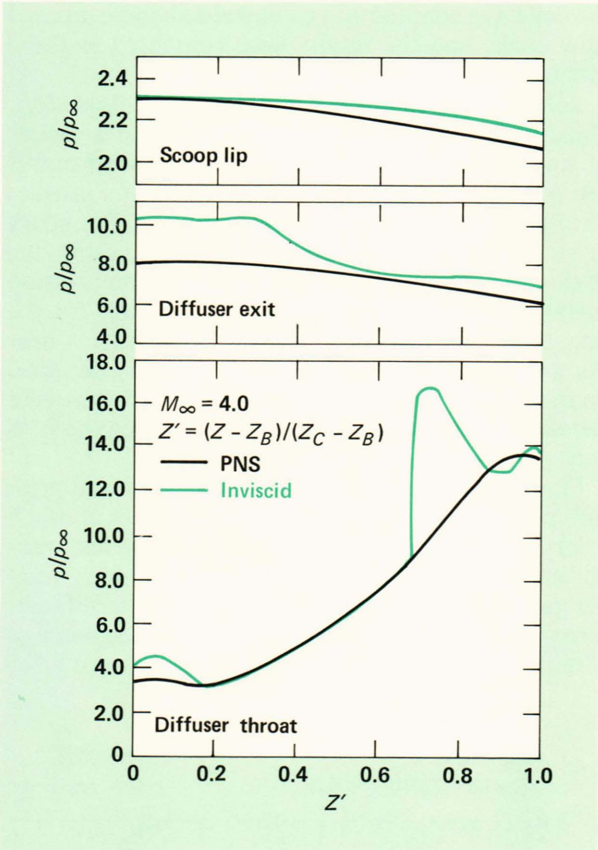


Figure 9 – Pressure profiles for an axisymmetric supersonic inlet internal flowfield.

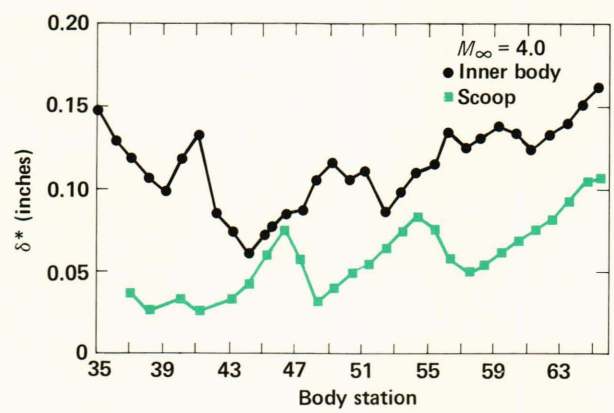


Figure 10 – Boundary layer displacement thickness distribution for the axisymmetric supersonic inlet internal flowfield.

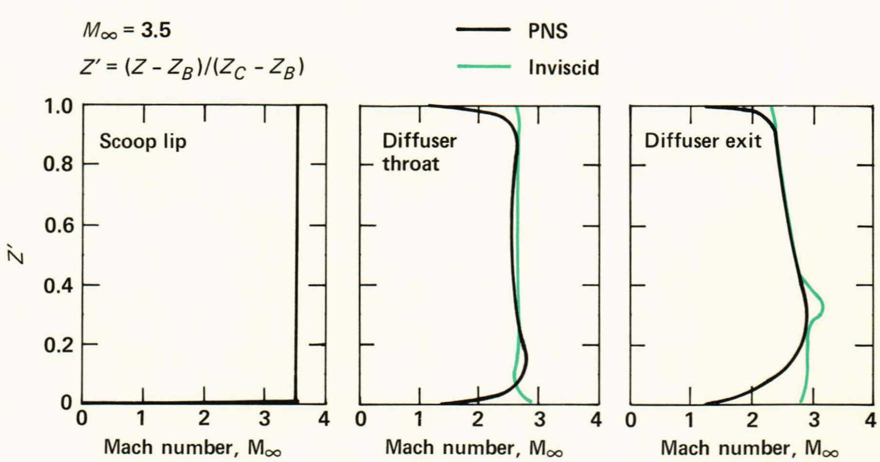


Figure 11 – Mach number profiles for the two-dimensional supersonic inlet internal flowfield.

compensate for the "mass defect." A model based on these computations is in the fabrication phase.

Similar solutions have been generated for a simplified two-dimensional model being used to examine inlet starting characteristics. Comparisons at Mach 3.5 between the PNS code and the inviscid code for this geometry are shown at three stations in Figs. 11 and 12, where Mach number and pressure profiles are plotted at the cowl lip, throat, and exit. Agreement is again good. Wind-tunnel tests have been conducted for this simplified inlet and agreement between the PNS results and experiment is excellent.

As discussed previously, an important feature of the viscous flow analysis is the ability to obtain a quantitative assessment of the effects of viscosity on inlet efficiency. Figure 13 shows this effect as a function of axial station for both the axisymmetric and two-dimensional inlet designs. The ratio of the kinetic energy efficiency obtained from PNS and inviscid flowfield analyses is compared, where for reference the η_{KE} values at the combustor entrance (exit axial station) for inviscid flow are 0.991 and 0.984 for the two-dimensional and axisymmetric designs, respectively. Thus, viscous losses comprise about one-half of the total loss in typical hypersonic inlet designs examined thus far.

CONCLUSIONS

The advent of very-high-speed computers provides an opportunity to use computational techniques for the design of airbreathing propulsion systems. Judicious choices must be made to obtain results that are

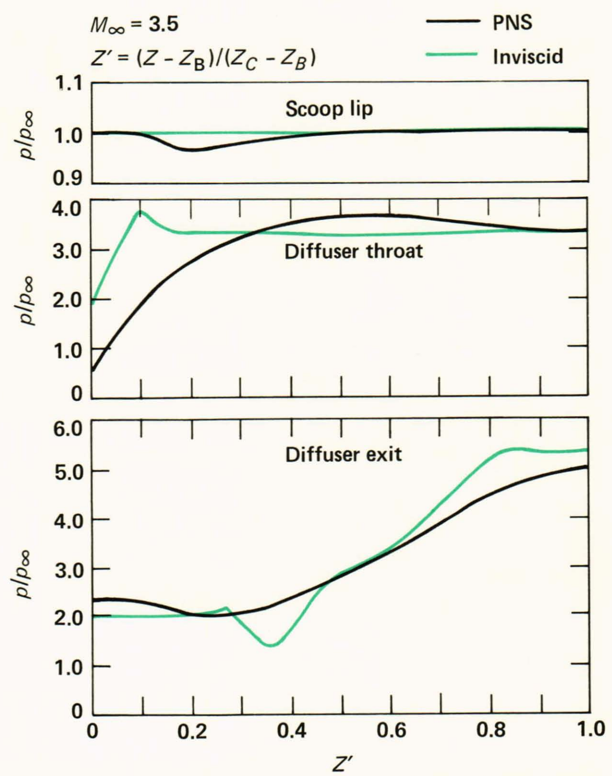


Figure 12 – Pressure profiles for the two-dimensional supersonic inlet internal flowfield.

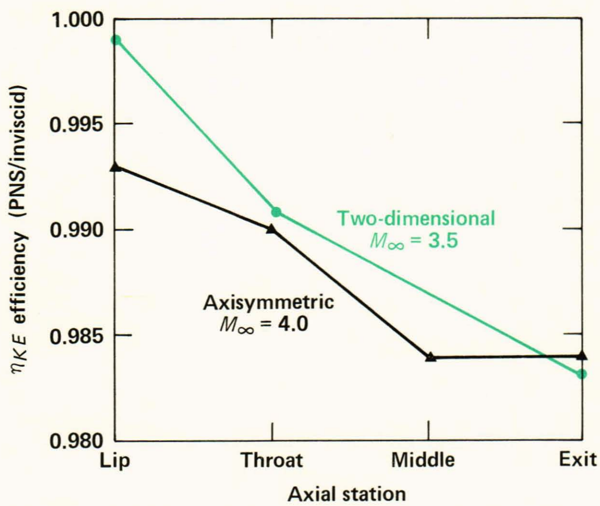


Figure 13 – Viscous effects on internal efficiency.

sufficiently accurate to be useful in developing designs and at the same time not be prohibitively expensive. At present, the techniques described herein have proven to be extremely useful in both eliminating designs with poor performance potential and identifying promising designs to be examined in wind-tunnel tests. Much work must be done to realize the full potential of this approach.

REFERENCES

[1]D. R. Chapman "Computational Aerodynamics Development and Outlook," *AIAA J.* **17**, 1293-1313, (Dec 1979).

[2]R. M. Beam and R. F. Warming, "An Implicit Factored Scheme for the Compressible Navier-Stokes Equations," *AIAA J.* **16**, 393-402 (Apr 1978).

[3]R. W. MacCormack, "A Numerical Method for Solving the Equations of Compressible Viscous Flow," *AIAA J.* **20**, 1275-1281 (Sep 1982).

[4]S. M. Ulam, *The Adventures of a Mathematician,* Charles Scribner's Sons, New York (1976).

[5]P. J. Roache, *Computational Fluid Dynamics,* Hermosa Press, Albuquerque (1971).

[6]J. P. Drummond and E. H. Weidner, "Numerical Study of a Scramjet Engine Flowfield," *AIAA J.* **20**, 1182-1187 (Sept 1982).

[7]D. D. Knight, "Improved Calculation of High-Speed Inlet Flows," *AIAA J.* **19**, 172-179 (Feb 1981).

[8]A. B. Wardlaw, J. M. Solomon, and F. P. Baltakis, "Supersonic Inviscid Flowfield Computations of Missile Type Bodies," *AIAA J.* **19**, 899-906 (Jul 1981).

[9]J. M. Solomon, M. Ciment, R. E. Ferguson, and J. B. Bell, "Inviscid Flowfield Calculations for Re-entry Vehicles with Control Surfaces," *AIAA J.* **15**, 1742-1749 (Dec 1977).

[10]F. S. Billig, P. J. Waltrup, and R. D. Stockbridge, "Integral Rocket Dual-Combustion Ramjets: A New Propulsion Concept," *AIAA J. Spacecraft and Rockets* **17**, 416-423 (Sep-Oct 1980).

[11]J. A. Schetz, F. S. Billig, and S. Favin, "Flowfield Analysis of a Scramjet Combustor with a Coaxial Fuel Jet," *AIAA J.* **20**, 1268-1274 (Sep 1982).

[12]M. D. Griffin, "Calculation of Inviscid Air Capture and Additive Drag for 3-D Supersonic Inlet Flow," *Proc. 1983 JANAF Propulsion Meeting,* Monterey, Calif., (14-18 Feb 1983).

[13]L. B. Schiff and J. L. Steger, "Numerical Simulation of Steady Supersonic Viscous Flow," *AIAA J.* **18**, 1421-1430 (Dec 1980).

[14]W. S. Helliwell, R. P. Dickinson, and S. C. Lubard, "Viscous Flow Over Arbitrary Geometries at High Angle of Attack," *AIAA J.* **19**, 191-197 (Feb 1981).

[15]M. M. Rai and D. S. Chaussee, "New Implicit Boundary Procedures: Theory and Application," AIAA Paper 83-0123, AIAA Aerospace Sciences Meeting, Reno (10-13 Jan 1983).

[16]M. D. Griffin, F. S. Billig, and M. E. White, "Applications of Computation Techniques in the Design of Ramjet Engines," 6th International Symposium on Air Breathing Engines, 16th International Aeronautical Congress, Paris, (6-10 Jun 1983).